Journal of Engineering Physics and Thermophysics, Vol. 75, No. 1, 2002

NUMERICAL MODELING OF LAMINAR CIRCULATION FLOW IN A SQUARE CAVITY WITH A MOVING BOUNDARY AT HIGH REYNOLDS NUMBERS

S. A. Isaev,[a] A. G. Sudakov,[a]
N. N. Luchko,[b] and T. V. Sidorovich[b]

UDC 532.517.2

Laminar circulation flow in a square cavity is numerically analyzed at high Reynolds numbers (of up to 40,000).

1. Numerical modeling continues to be a vigorously developing area in hydromechanics, first of all due to the continuous progress being made in computer technologies. Noteworthy is its significance as a universal tool for investigations, i.e., wide distribution in the form of numerous program products, including commercial ones. At present, a numerical experiment supplements (or rather anticipates) a physical one and sometimes replaces it. By and large, it may be stated that computer modeling has gradually come into the industrial stage of its development [1].

Nonetheless, methodological investigations continue to arouse interest (just as thirty years ago). Traditionally, such works play the role of a testing ground for checking new methodological concepts and estimating the adequacy of the results obtained with the instruments designed on their basis. Their importance is also explained by the fact that today, in contrast to past years, users give too much credence to numerical modeling, but only thorough testing of numerical tools used in solving model problems can reveal "cracks" contained in them.

In this respect, one of the problems simplest in formulation and most economical of computational resources is the problem of circulation flow of a viscous incompressible fluid in a square cavity, which represents an unusual kind of "mirror" of the history of numerical modeling. It has been solved many times, especially for the purpose of improving the methods of solution of difference equations and algorithms of computational schemes [2].

2. Over several decades, the problem of circulation flow of a viscous incompressible fluid in a plane square cavity with a moving upper cover has attracted interest as a test problem [2, 3]. This interest has also been stimulated by the fact that the flow considered possesses a set of structural features characteristic of detached (separating) flows. The problem imposes very low requirements upon the resources of computers (which was very important at the initial stage of development of numerical hydrodynamics, because at that time the resources of computers were extremely limited) because a flow can be localized in a square computational region, which advantageously differentiates this problem from other test problems, such as the problem of flow along a cylinder, for whose calculation it is necessary to use a large region. There is also no need to establish the conditions at the boundaries of the computational region (adhesion condition) in contrast, for example, to the modeling of flow along a cylinder in the case where the solution is dependent on the conditions at the boundaries of the computational region.

---

[a]Academy of Civil Aviation, St. Petersburg, Russia; email: isaev@SI3612.spb.edu; [b]A. V. Luikov Heat and Mass Transfer Institute, National Academy of Sciences of Belarus, Minsk; email: lusid@hmti.ac.by.
Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 75, No. 1, pp. 54–60, January–February, 2002. Original article submitted April 27, 2001.

To save the required computer memory, one initially used explicit schemes of solution of the Navier–Stokes equations written in vorticity–stream function variables [3]. In this case, the derivatives in the equations of vortex transfer and in the equation of connection of the vorticity and the stream function were represented with the use of central differences. The computational region was covered by a uniform grid with a fairly large step, and the boundary condition for the vorticity on a solid wall had the first order of approximation (Thom condition). The steady-state solutions of the problem have been obtained only at low (lower than 120) [4] and moderate [5] Reynolds numbers, since the increase in Re leads to a computational instability characteristic of central-difference schemes. The use of the relaxational method for solving the difference equations with very low relaxation coefficients [6] made it possible to somewhat extend the range of variation of the Reynolds number to 700, but in this case, the results of calculation of the flow for moderate (of the order of 400 or higher) Reynolds numbers were insufficiently accurate.

The use of the alternating directions method to solve unsteady Navier–Stokes equations was more successful [7]. Such an approach made it possible to obtain steady-state solutions for Reynolds numbers of up to 1000, but the quality of the modeling of the flow was low because of, first of all, the insufficient number of computational points (nodes). For example, for uniform grids with the number of points 20×17 and 39×33, at Re = 500 there were significant structural differences in solutions obtained on the fine and coarse large grids. A "pseudosolution" of the problem was obtained for the first time with the use of a coarse grid: two large-scale vortices in a square cavity, which were then observed in [8, 9]. Thus, the first numerical investigations of circulation flows have revealed the difficulties associated with obtaining a convergent solution of the problem and pointed to the comparatively low accuracy of the results, which rapidly decreases with increase in the Reynolds number.

It has been possible to improve the convergence of the computational procedure when the one-sided upwind differences were used in representation of the convective terms of the transfer equations. The difference schemes of first order of approximation and, in particular, the Spalding integro-interpolation scheme allowed one to carry out calculations of circulation flows at Reynolds numbers as high as is wished [8, 10]. However, the great numerical diffusion introduced in this case and caused by the errors in approximation of the initial equations shades the processes of diffusion transfer which are caused by molecular viscosity and distorts the solution of the problem beginning with Re = 300 [11]. It has been established that the results of calculations essentially depend on the number of computational points, their positions, and the order of approximation of the boundary condition for the vorticity [10].

The unsatisfactory quality of the modeling of circulation flow at moderate and high Reynolds numbers, characteristic of all the works carried out, stimulated the perfection of computational algorithms, first of all, in the direction of increasing the order of approximation of the difference analogs of the convective terms of the Navier–Stokes equations and application of nonuniform grids with bunching of points in the zones of large gradients or uniform grids with a very small step. Realization of the above concept made it possible to obtain sufficiently exact solutions for high (of the order of 10^4) Reynolds numbers at the end of the 1970s and at the beginning of the 1980s. In this case, different approaches were used: (a) combination of the alternating directions method with the space-varying coefficients of relaxation and grids with bunching of points toward the walls in accordance with the trigonometric law [12]; (b) combination of the Adams–Bashfort method for representation of the unsteady term with approximation of the convective terms of the vortex-transfer equation by the Arakawa schemes of second and fourth order and nonuniform computational grids [13, 14]; (c) multigrid method realized on uniform embedded grids with a number of points of up to 257×257 [15]; (d) Agarval method of third order of upwind approximation, realized on uniform grids with a number of points of up to 121×121 [16]; (e) combination of the Newton method for systems of nonlinear equations and of the procedure of continuation of solution in the case where the Reynolds number changes up to 104, realized on uniform grids with a number of points of up to 181×181 [9].

All the above works have been carried out with the Navier–Stokes equations written in transformed variables with the use of consistent grids; the grid points were positioned on the walls of the cavity. A much smaller number of results have been obtained for the problem considered when the Navier–Stokes equations written in natural variables were used. In [17, 18] (beginning of the 1980s), different modifications of the Leonard scheme in combination with the SIMPLE algorithm were analyzed and the flow in a cavity was calculated at Reynolds numbers of up to Re = 1000. The multigrid method proposed by Brandt in [19] was extended to the solution of the Navier–Stokes equations in natural variables on staggered grids with the number of points 321×321 at Re numbers of up to 5000. However, the use a hybrid scheme for approximation of the convective terms of the momentum equations did not give sufficiently accurate results at high (as high as 2000 or higher) Reynolds numbers.

Of special note are methodological investigations associated with supercomputer-aided testing of problems. Thus, in [20], the Navier–Stokes equations written in transformed variables vorticity–velocity components were solved on staggered grids with the number of points of the order of 100×100 using the implicit alternating directions procedure written in completely vectorized form. The Reynolds number was varied from 1000 to 5000. The results of calculations carried out on different supercomputers with vector and scalar processors were compared. In [21], the unsteady Navier–Stokes equations written in natural variables were integrated with a supercomputer by the explicit two-layer scheme of Adams and Bashfort; the modified Leonard scheme (QUICKMAC) was used for approximation of the convective terms. The steady-state solution of the problem, beginning with the zero initial conditions, has been obtained on a highly nonuniform grid with the number of points 44×44 at Re = 10^4 in 140,000 time steps with $\Delta t = 0.001$.

The calculated and experimental results of investigations of circulation movement of a fluid in a square cavity with a moving boundary have been generalized in [2]. A computational algorithm based on the implicit factorized method of solution of the Navier–Stokes equations written in natural variables and on the use of the Leonard scheme for approximation of the convective terms on the explicit side of the equation is presented. In essence, this is a prototype of the algorithm developed in this work. In the same work, the original results of calculations of the flow on a uniform grid with the number of cells up to 40×40 at Reynolds numbers varying from 100 to 5000 are presented.

Thus, the solution of the problem of circulation motion of a fluid in a cavity illustrates the history of development of computational aerodynamics. The works considered do not exhaust the complete list of numerical investigations of such detached flows. There is no doubt that to them we should add the works of the last decade (see, for example, [22–24]). Nonetheless, it is obvious that computational material tremendous in volume and various in content has been accumulated. It can be used for analysis of different computational algorithms and schemes of solution of the Navier–Stokes equations and for investigation of the physical nature of detached flows.

Systematization and analysis of the results obtained allowed us to propose several parameters that can be used as criteria for evaluating the quality of the discrete model used; these parameters include the maximum value of the stream function ψ_m in the computational region and the vertical and horizontal dimensions of the secondary corner vortices developed as the Reynolds number increases. As has been noted in [2, 14], the quantity ψ_m has a clear physical meaning (it determines the intensity of the circulation flow in the cavity). As distinguished from the local parameters of the flow, such as the extremum value of the friction at the moving boundary, ψ_m allows one to characterize the flow in the cavity as a whole. The experience gained from the calculations [2, 6, 10] carried out suggests that ψ_m essentially depends on the scheme factor: the grid step, the arrangement of the points in the computational region, and the form of the boundary conditions. It is quite evident that the final solution of the problem can be a variant of calculation which is independent of the above scheme factors.

In the present investigation, prominence is given to the modeling of circulation flow of a viscous incompressible fluid in a square cavity at high Reynolds numbers (higher than 10^4). The range of variation

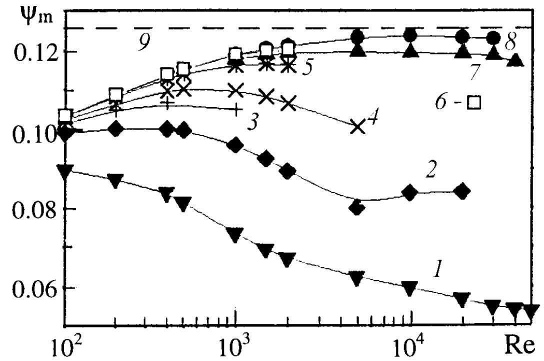

Fig. 1. Dependence of the predicted intensity of the circulation flow in a cavity on the Reynolds number and the number of computational cells in the case of their uniform distribution [1) 10 × 10; 2) 20 × 20; 3) 30 × 30; 4) 40 × 40; 5) 80 × 80; 6) 200 × 200] and when they are arranged with bunching in the neighborhood of the wall (the minimum step is $5 \cdot 10^{-4}$) [7) 100 × 100; 8) 200 × 200; 9) asymptotic (for Re $\to \infty$) Burggraf solution].

of Re numbers considered has practically not been investigated. Moreover, the computational resources of modern personal computers offer possibilities for solving the problem on very narrow grids with the required concentration of near-wall cells to provide sufficient accuracy.

3. The computational methodology used in the present work is based on solution of the Navier–Stokes equations in natural variables by the finite-volume method in the context of the concept of splitting by physical processes. The approach, which has been described for the first time in [2], tested in [22], and further developed in [25, 26], has several characteristic features: (a) representation of the initial steady-state equations in increments of dependent variables and carrying out the finite-volume approximation of the equations using a centered computational template that is based on the assignment of dependent variables at the center of the computational cell of a structured curvilinear grid, (b) approximation of the convective terms of equations on the implicit side by the upwind difference scheme with quadratic interpolation, proposed by Leonard, (c) introduction of additional numerical diffusion to efficiently smooth the high-frequency oscillations occurring in the calculations at high Reynolds numbers, (d) SIMPLEC procedure of pressure correction, (e) solution of difference equations by the method of incomplete matrix factorization, and (f) orientation to personal computers.

4. To calculate the flow in the cavity use is made of grids containing 10 × 10, 20 × 20, 40 × 40, 80 × 80, 100 × 100, and 200 × 200 cells with a uniform and nonuniform distribution within the region. The Reynolds numbers are selected to be 100, 200, 400, 500, 10^3, 1500, 2000, 5000, 10^4, $2 \cdot 10^4$, $3 \cdot 10^4$, and $4 \cdot 10^4$. The minimum near-wall step is $5 \cdot 10^{-4}$.

The results of the calculations of the intensity of the circulation flow in the cavity, carried out on grids with a different number and distribution of cells, are summarized in Fig. 1. It is evident that the curves of the dependences $\psi_m(\mathrm{Re})$ are stratified when the density of the cells in the region decreases. This is quite justified, since for uniform and not very narrow grids the error of determination of the boundary conditions on the moving wall increases with increase in the Reynolds number. Moreover, the reproduction of the primary large-scale vortex arising in the cavity also requires the proper density of points in its central zone. As is seen from the results presented, at the high Reynolds numbers considered the required independence of the solution from the number and distribution of the cells in the region is practically attained for fairly narrow grids (100 × 100 and 200 × 200).

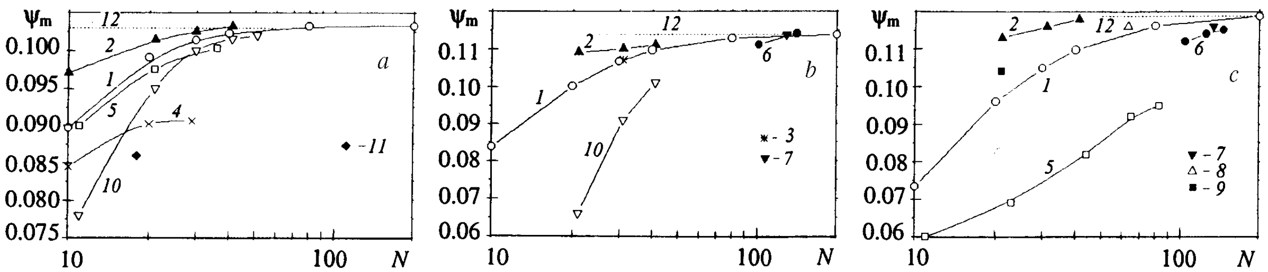

Fig. 2. Dependence of ψ_m on the number of points of a uniform grid: at Re = 100 (a), 400 (b), and 100 (c). The notation of the curves is presented in Table 1.

TABLE 1. Notation to the Results Presented in Fig. 2

Number of curves	Representation of equations	Difference scheme	Grid	Reference
1	*u–v–p*	QUICK	CG	This work
2	*u–v–p*	QUICK	SG	[2]
3	*u–v–p*	QUICKER	SG	[18]
4	*u–v–p*	UDS	SG	[2]
5	ω–φ	UDS	CG	[10]
6	ω–φ	CDS	CG	[9]
7	ω–φ	CDS+UDS	CG	[15]
8	ω–φ	ADS	CG	[16]
9	ω–φ	Arakawa scheme	CG	[13]
10	ω–φ	CDS	CG	[6]
11	ω–φ	CDS	CG	[7]
12	Asymptotic quantities			[2]

Figure 2 shows results obtained by different difference methods on different grids at Reynolds numbers of 100, 400, and 1000 versus the number of points N in the computational region. The references used are summarized in Table 1. As follows from Fig. 2, the results of the numerical modeling of circulation flows largely depend on the number of grid points in the case where calculations are carried out on coarse grids, which has already been noted in the analysis of Fig. 1. At the same time, there is an asymptotics in the behavior of the curves $\psi_m(N)$, which is attained for moderate Reynolds numbers even on grids that are not so narrow: at a Reynolds number of 100, 400 points in the computational region are quite sufficient to obtain results close to the asymptotic ones (on condition that a staggered grid (SG) and a difference scheme of second order of approximation are used). It is of interest to compare the results of calculation of the flow in the cavity by different methods on fairly coarse practical grids at Re = 100, since for such grids (of the order of 40×40) all the algorithms considered give the same results. Thus, on a consistent uniform grid (CUG) with the number of points 20×20 the central-difference scheme (CDS) in combination with the Thom boundary condition for the vorticity [3], the upwind difference scheme (UDS) in combination with the boundary condition for the vortex of second order of approximation (Woods condition), and the QUICK scheme proposed by Leonard give approximately the same results for the quantity ψ_m independently of the form of representation of the Navier–Stokes equations. The use of the boundary conditions of higher order of approximation improves the quality of modeling of circulation flow. It should be noted that for this type of flow maintained by the friction stress on a moving wall, discretization of the boundary conditions at low Reynolds numbers is of somewhat greater importance than the order of approximation of the scheme.

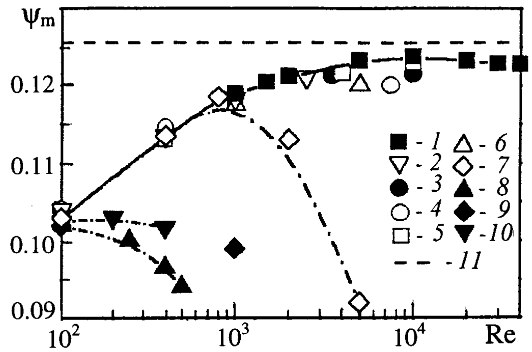

Fig. 3. Dependence of ψ_m on the Reynolds number for different algorithms of calculation of the flow in a cavity. The notation of the curves is presented in Table 2.

TABLE 2. Notation to the Results Presented in Fig. 3

Number of curves	Representation of equations	Difference scheme	Grid	Reference
1	*u–v–p*	QUICK	CG	This work
2	*u–v–p*	QUICK	SG	[2]
3	ω–φ	CDS	CG	[12]
4	ω–φ	CDS+UDS	CG	[15]
5	ω–φ	CDS	CG	[9]
6	ω–φ	ADS	CG	[16]
7	*u–v–p*	HDS	SG	[19]
8	ω–φ	CDS	CG	[7]
9	ω–φ	UDS	CG	[10]
10	ω–φ	CDS	CG	[6]
11	Asymptotics according to Batchelor			[6]

As Re increases, the curves $\psi_m(N)$ in Fig. 2 also undergo a marked stratification, which points to the advantage of the implicit algorithm described here, which is based on the use of the Leonard scheme as well as staggered and consistent grids. The analysis of Fig. 2b and c shows that the results obtained on fairly economic grids are close to the asymptotic results obtained on very narrow grids (100 × 100 or more points in the computational region). It is of interest to note that the results of solution of the Navier–Stokes equations on a consistent grid are exceeded in accuracy by the results of calculations on a staggered grid. However, the results of calculations by the QUICKER scheme are practically coincident with the results obtained in the present work.

Bunching of the grid points in the neighborhood of the walls allows one to model the near-wall gradient flows more correctly. It was established that it is necessary to transform a grid to calculate flows at high Re. The redistribution of points in the computational region also makes it possible to use grids that are not so narrow in the central region to correctly describe a large-scale practically ideal vortex. As follows from [2], on a grid with the number of points 21 × 21 one can obtain a solution close to the solution obtained on a grid containing 151 × 151 points (Re = 1000). We should note that in this case of importance is discretization of the convective terms by the schemes of high order of approximation (second or fourth) [13].

5. Some of the results obtained are shown in Figs. 3–6.

Figure 3 shows results of calculations of the quantities ψ_m as functions of the Reynolds number by different methods. To the figure taken from [2] we added a curve calculated on a nonuniform 200 × 200 grid. The fact that the initial values of the stream function, obtained by different methods, are grouped near

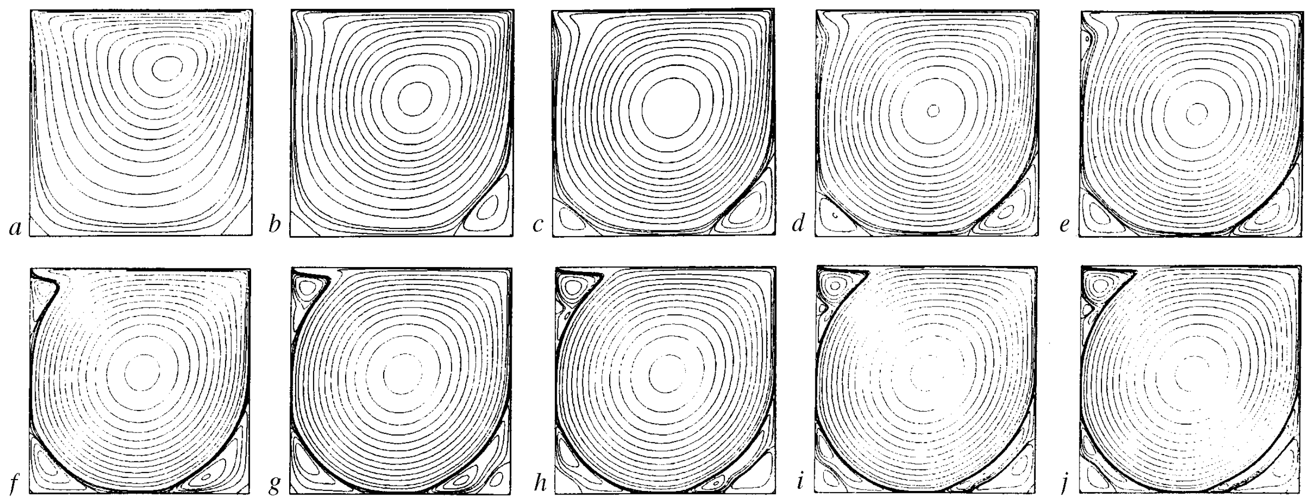

Fig. 4. Evolution of the pattern of flow in a square cavity with increase in the Reynolds number: a) Re = 100, b) 400, c) 10^3, d) $1.5{\cdot}10^3$, e) $2{\cdot}10^3$, f) $5{\cdot}10^3$, g) 10^4, h) $2{\cdot}10^4$, i) $3{\cdot}10^4$, and j) $4{\cdot}10^4$.

one curve with a spread no larger than 1.5–2% has engaged our attention. This points to the correctness of the dependences $\psi_m(Re)$, on the one hand, and a sufficiently high accuracy of the algorithm developed, on the other. Figure 3 also shows a line corresponding to the asymptotic approximation for the flow in a cavity in the case $Re \to \infty$ [6]. Thus, the tendency toward intensification of circulation motion of the fluid in a cavity as the Reynolds number increases turns out to be true; the function $\varphi_m(Re)$ is limited and tends to the limiting value of $\psi_m(Re \to \infty)$ obtained on the basis of the model of a large-scale ideal vortex fitted into a square region and separated from the walls by a thin viscous shear layer (Batchelor model [6]).

At the same time, the results shown in Fig. 3 by dash-dot lines point to the erroneous tendencies in the behavior of ψ_m, associated with the loss in accuracy of the numerical modeling of circulation flows. Indeed, schemes of second order of approximation [6, 7] in combination with the boundary condition of first order and uniform grids cannot describe sufficiently accurately the effects of viscous interaction at high Re (the thicknesses of the viscous boundary layers become comparable to the minimum step of the grid or smaller than it); therefore, with increase in Re, the connection between the moving boundary and the circulation flow in the vortex becomes weaker. For schemes of first order of approximation (or hybrid schemes, such as an HDS) even the use of a grid with steps nonuniform along the space coordinates [10] or a very fine grid (321×321) (in the case of the multigrid approach to the solution of the problem) [19] gives no positive results, since the mechanism of computational dissipation introduced into the calculation by these schemes (see [2] for details) is much stronger than the actually acting mechanism of physical viscosity. A consequence of the above factors is deceleration of the circulation flow in the cavity with increase in Re, which was observed in the calculations.

The evolution of the vortex structure with increase in Re, calculated on the 200×200 grid (Fig. 4), allowed us to reveal several phases in the development of the flow pattern. It should be emphasized that reciprocal motion of the fluid in a cavity is induced by the displacement of the moving cover and is maintained by the friction stress on it. In practice, flowing of the formed near-wall jet over the side wall of the cavity occurs. At low and moderate Reynolds numbers the viscous effects are dominant and their influence extends virtually to the entire region of the cavity. The secondary corner vortices arise as a result of adaptation of the turning flow to the rectangular configuration of the region. When the cover moves from left to right, at first the secondary vortex adjacent to the right wall of the cavity appears. This is a result of interaction of the reflected spreading near-wall jet with the bottom of the cavity. At the same time, the secondary

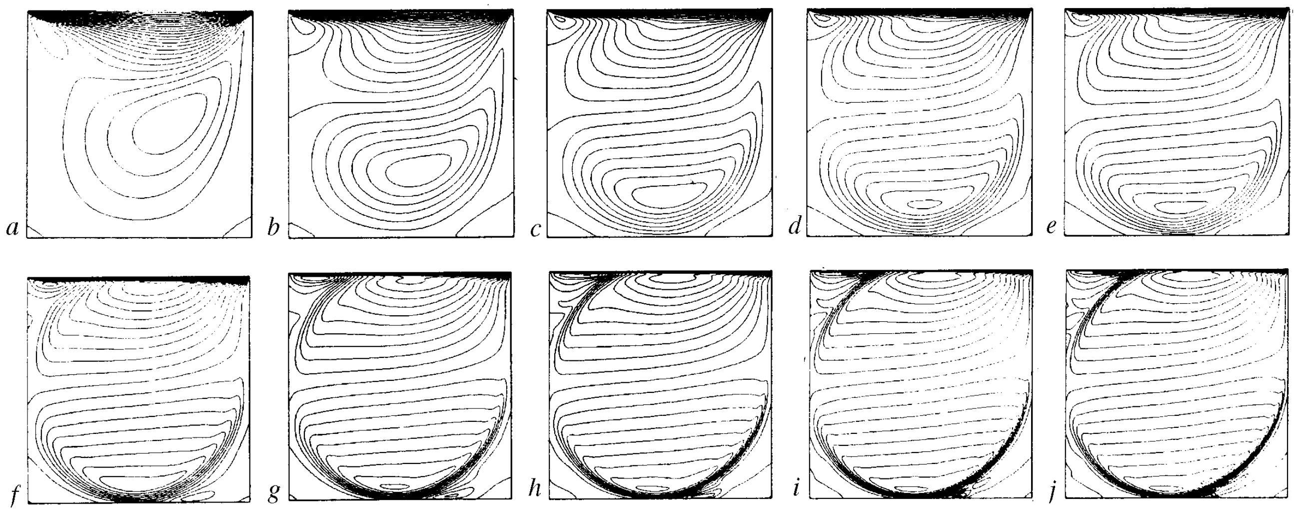

Fig. 5. Evolution of the patterns of isolines of the horizontal velocity component with increase in the Reynolds number (a–j): the values of Re are the same as in Fig. 4.

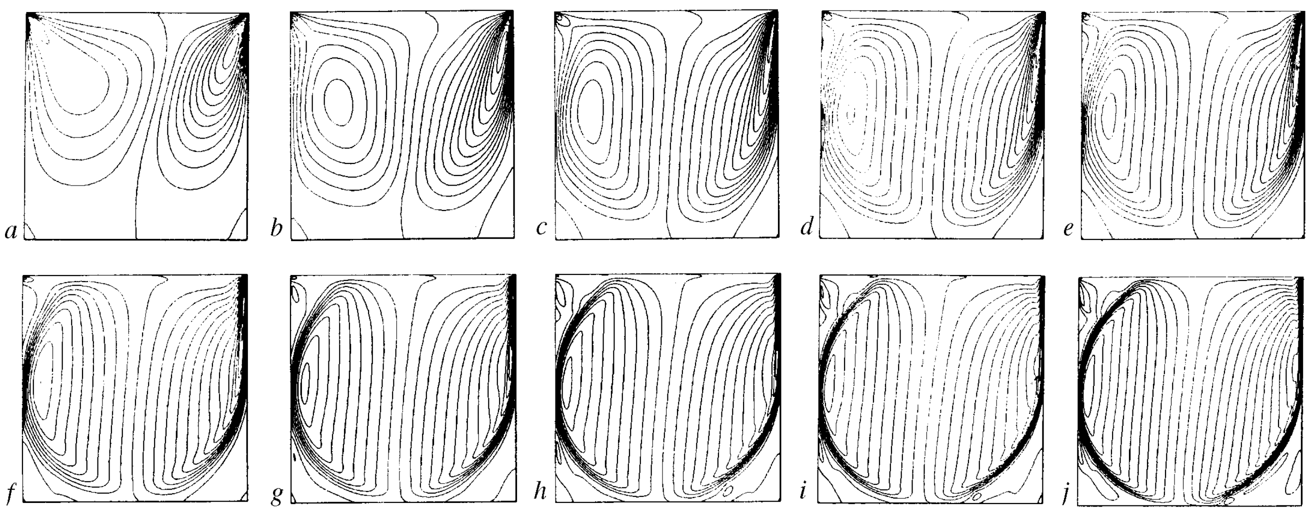

Fig. 6. Evolution of the patterns of isolines of the vertical velocity component with increase in the Reynolds number (a–j): the values of Re are the same as in Fig. 4.

vortex at the lower left corner of the cavity arises only when the intensity of the secondary flow in the cavity becomes significant, i.e., much later than that at the lower right corner.

As the Reynolds number increases, the influence of the viscous effects decreases and the effects of convective transfer become dominant. In the primary vortex, a core of constant vorticity appears, which, growing, gradually occupies the entire central part of the cavity and approaches its walls. However, the central vortex does not tend to occupy the entire region of the cavity, as is assumed in the limiting case (Re $\to \infty$) of the flow modeled in [6]. The dimensions and intensity of the secondary corner vortices also increase. Beginning with Re = 1500, one more vortex structure arises at the upper left corner. This structure is caused by the turning and braking of the flow as the moving cover is approached. The vortex pattern of the flow in a square cavity was determined to be such already in the middle 1970s.

The calculation results presented in Figs. 5 and 6 allow one to perform a more detailed analysis of changes in the pattern of detached flow in a cavity at high Reynolds numbers. It is clearly seen that all the structural elements of the flow tend to a gradual development as Re increases. Despite the fact that the topology of the flow in the primary vortex remains unchanged, in practice (approximately from Re = 2000), the gradients in the velocity field increase because of the progressing concentration of the isolines of the velocity components in the neighborhood of the walls and the increase in the extremum values of the surface distributions of the friction coefficients.

At high Reynolds numbers, the zone of mixing between the thin near-wall layer and the large-scale vortex is formed near the moving boundary. At moderate and average Reynolds numbers, as has already been noted in [2], this zone is not there because of, first, the large thickness of the near-wall layers and, second, the absence of the secondary vortex near the moving wall. The zigzag shape of the profile of the longitudinal velocity component at high Re reflects the process of mixing of two flows different in energy characteristics: the nonuniform near-wall flow entrained by the moving wall of the fluid and the flow circulating in the large-scale vortex. Thus, as Re increases, the flow in the primary vortex and in the secondary vortices is intensified with increase in their dimensions. The influence of the viscous effects is localized in the small neighborhood of the walls and in the corner zones.

Analysis of the results of the calculation of the flow at Re varying from 10,000 to 40,000 shows that the process of extension of the secondary vortices is intensified and leads to their pinching and subdivision, i.e., we have multiple breaking of the secondary vortices due to the intensification of the near-wall vortex formations which both are located directly in the corner zones and develop inside the vortex adjacent to the upper corner of the cavity. We can note a tendency toward comminution of the secondary vortex structures by breaking them into small-scale vortex cores as the Reynolds number increases further.

This work was carried out with financial support from the Russian Foundation for Basic Research (project Nos. 00-02-81045 and 99-01-00722) and the Belarusian Republic Foundation for Basic Research (project No. F99R-104).

NOTATION

u and v, Cartesian velocity components; p, pressure; ω, vorticity; ψ, stream function; Re, Reynolds number; N, number of points along the coordinate direction. Subscript: m, minimum value.

REFERENCES

1. A. D. Gosman, *Trans. IChE*, **76**, Pt. A, No. 2, 153–161 (1998).
2. I. A. Belov, S. A. Isaev, and V. A. Korobkov, *Problems and Methods of Calculation of Separating Flows of an Incompressible Fluid* [in Russian], Leningrad (1989).
3. P. J. Roache, *Computational Fluid Dynamics* [Russian translation], Moscow (1980).
4. M. Kawaguti, *J. Phys. Soc. Jpn.*, **16**, No. 1, 2307–2331 (1961).
5. L. M. Simuni, *Inzh. Zh.*, **4**, Issue 3, 446–450 (1964).
6. O. R. Burggraf, *J. Fluid. Mech.*, **24**, Pt. 2, 113–151 (1966).
7. V. M. Paskonov, V. I. Polezhaev, and L. A. Chudov, *Numerical Modeling of Heat and Mass Transfer Processes* [in Russian], Moscow (1984).
8. J. D. Boseman and C. Dalton, *J. Comput. Phys.*, **12**, No. 3, 348–363 (1973).
9. R. S. Schreiber and H. B. Keller, *J. Comput. Phys.*, **49**, No. 3, 310–333 (1983).
10. A. D. Gosman, W. M. Pun, A. K. Runchal, et al., *Numerical Methods of Study of Viscous Fluid* [Russian translation], Moscow (1972).
11. D. de Vahl and G. D. Mallinson, *Computer Fluids*, **4**, No. 1, 29–43 (1979).
12. A. S. Benjamin and V. E. Denny, *J. Comput. Phys.*, **33**, No. 3, 340–358 (1979).

13. I. A. Belov and S. A. Isaev, *Zh. Prikl. Mekh. Tekh.*, No. 1, 41–45 (1982).
14. S. A. Isaev, *Chisl. Met. Mekh. Sploshn. Sred*, **14**, No. 5, 70–78 (1983).
15. U. Chia, K. N. Chia, and C. T. Shih, *J. Comput. Phys.*, **48**, No. 2, 387–411 (1982).
16. R. K. Agarval, *A Third-Order-Accurate Upwind Scheme at High Reynolds Numbers*, AIAA Paper No. 81-0112 (1981).
17. T. Han, J. A. C. Humphrey, and B. E. Launder, *Comput. Meth. Appl. Mech. Eng.*, **29**, No. 1, 81–95 (1981).
18. A. Pollard and A. L.-W. Siu, *Comput. Meth. Appl. Mech. Eng.*, **35**, No. 3, 293–313 (1982).
19. S. P. Vanka and G. K. Leaf, *An Efficient Finite-Difference Calculation Procedure for Multidimensional Fluid Flows*, AIAA Paper No. 84-1244 (1984).
20. G. Guj and F. Stella, *Int. J. Numer. Meth. Fluids*, No. 8, 405–416 (1988).
21. Y. Nakamura and Y. Takemoto, in: *Proc. Int. Symp. Fluid Dyn.*, Vol. 1, Tokyo (1985), pp. 386–397.
22. S. A. Isaev and A. E. Usachov, in: *Collection of Papers "Industrial Aerodynamics"* [in Russian], Issue 4 (36), Moscow (1991), pp. 43–75.
23. P. N. Vabishchevich, A. N. Pavlov, and A. G. Churbanov, *Mat. Model.*, **8**, No. 7, 81–108 (1996).
24. P. N. Vabishchevich, A. N. Pavlov, and A. G. Churbanov, *Mat. Model.*, **9**, No. 4, 86–114 (1997).
25. S. A. Isaev, A. N. Kudryavtsev, and A. G. Sudakov, *Inzh.-Fiz. Zh.*, **71**, No. 4, 618–631 (1998).
26. S. A. Isaev, P. A. Baranov, N. N. Luchko, et al., *Numerical Modeling of the Separating Flow of an Incompressible Fluid in Square and Cubic Cavities with a Moving Boundary*, Preprint No. 7 of the Heat and Mass Transfer Institute, National Academy of Sciences of Belarus [in Russian], Minsk (1999).